Fracture problems with ANSYS
A (very) brief introduction

Introduction and objective

ANSYS is a commercial finite element code used in industry to solve large scale problems. It can be used to solve a wide variety of problems including linear and nonlinear structural response, buckling, modal analysis, full harmonic response, transient dynamic response, heat transfer, electro-magnetic and fluid flow problems.

ANSYS offers a large library of elements ranging from the simplest (1-D elastic bar element) to the very complicated (3-D nonlinear elasto-plastic element). Information about these elements and the type of analysis available in ANSYS can be found in various sources and manuals. Help is also available interactively within ANSYS.

The objective of this short exercise is to extract the value of the stress intensity factor for a compact tension test (CTT) linearly elastic specimen described in Section A4 of the E 3999 ASTM Standard (see first two pages of the Appendix of this document). Basically, we want to perform a 2-D structural analysis with ANSYS to extract in an automatic fashion the value of the stress intensity factor for various crack lengths and reproduce the results indicated in Equation (A4.1) and Table A4.5.3.1. This will allow us to assess the precision of the finite element code for fracture problems.

We will use the following utilities available in ANSYS

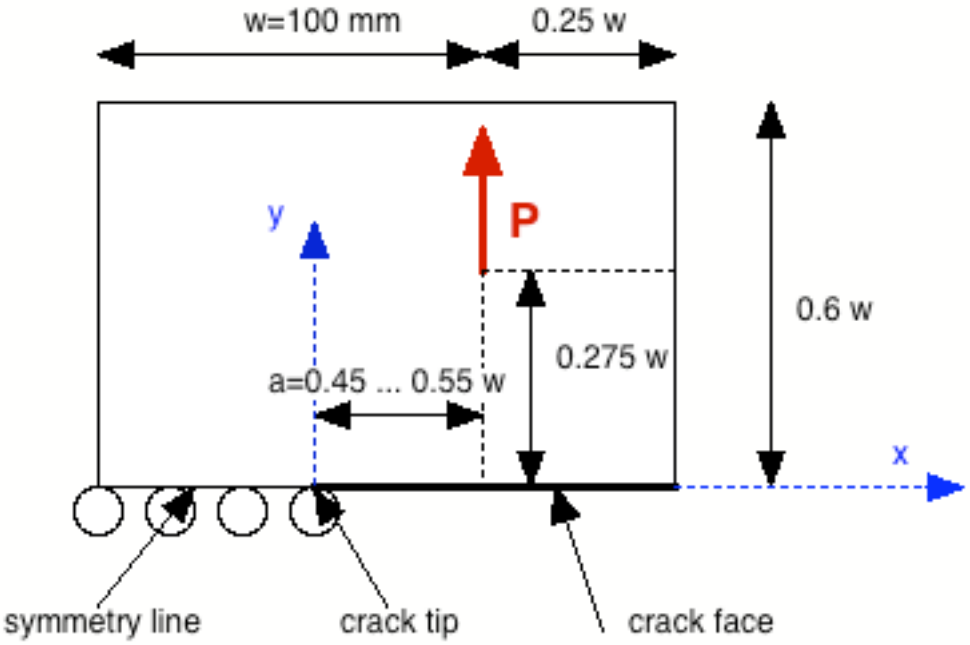
- log files used to run similar simulations automatically,
- scalar parameters definition, to automatically change the specimen size *w* and the crack length *a*,
- the **KSCON** command, which allows to generate focused mesh at the crack tip,
- the "skewed element" option, used to generate singular elements at the crack tip
- the **KCALC** command used to extract the value of the stress intensity factors

The section of the ANSYS Procedures Manual relative to the simulation of fracture problems is included in the appendix of this document. Please spend a few minutes reviewing the information relative to 2-D fracture analyses on pages 3-156 to 3-160.

The following lines describe the series of commands needed to solve the problem with ANSYS. The pick commands are denoted by **regular bold** words, while values to be entered (using the keyboard) are given in ***italic bold***.

Problem description

The structural problem to be solved is described in Figure A4.1 of the Appendix. Taking advantage of symmetry and simplifying the geometry a little bit, we will actually solve the following problem:



To prevent rigid body translation, we will fix the x-displacement of the point of application of the load P. To facilitate the creation of the focused mesh at the crack tip, we will place the origin of the axis system at the crack tip.

The material properties will be chosen as those of PMMA ($E = 4000\ N/mm^2$ and $\nu = 0.3$), although, for this problem, the material properties do not enter the expression of the stress intensity factor. The load P will be chosen as unity. We will perform the analysis in plane strain.

ANSYS analysis

This session assumes some familiarity with ANSYS. Only the steps specific to the fracture analysis will be described here. The other steps are identical to those of a conventional plane strain structural analysis.

1) Preliminary steps

Specify new job name (optional) and new title (optional).
Under the heading Parameters, define the two scalar parameters

w=100
a=0.45*w

Select Structural under Preferences… (optional)

2) Preprocessing step

Preprocessor >

2.1) Element and material definition

We will use 6-node triangular elements (Plane2). Make sure to select the plane strain option.
No real constant definition is needed for this type of element
Define a material set (Constant-Isotropic) with the appropriate properties (stiffness and Poisson's ratio).

2.2) Geometry definition

The easiest way to define the geometry is to define

- a semi-circle of radius *a/5* centered at the origin
- a large rectangle (xmin=a-w, ymin=0, xmax=a+w/4, ymax=0.6*w)
- a small rectangle (xmin=a, xmax=a+w/4, ymin=0.275*w, ymax=0.6*w

Then use the **Operate/Overlap/Areas** action to subtract the two smaller surfaces from the large rectangle (Figure 1).

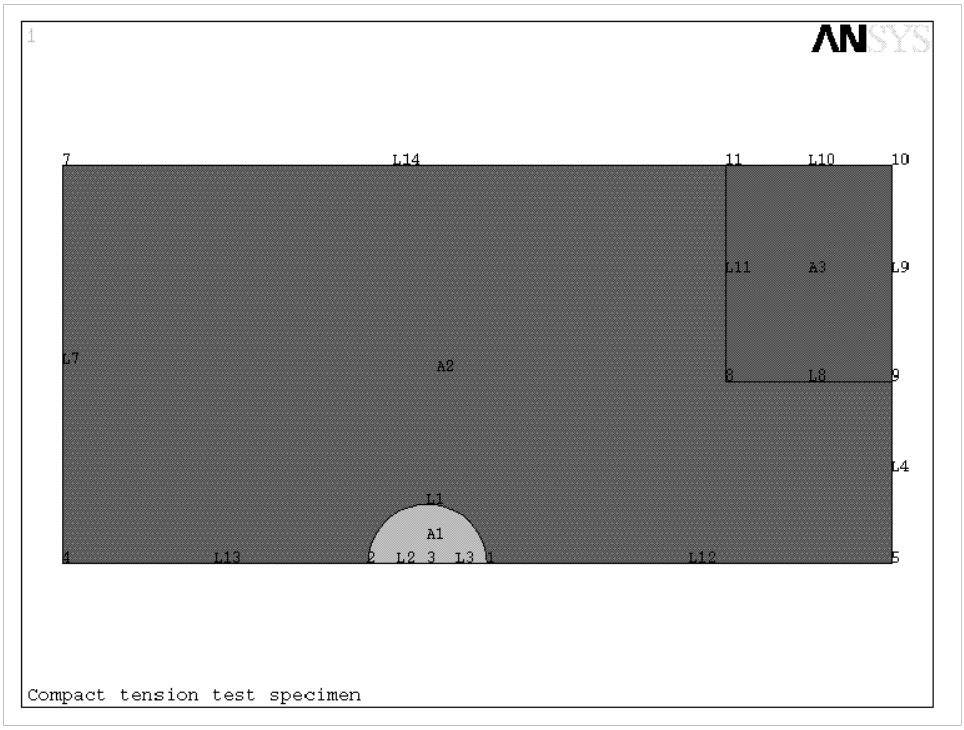


Figure 1. Definition of areas, lines and keypoints.

2.3) Mesh generation

First create the mesh in the semi-circle:

- First, create a concentrated mesh at the crack tip with **Mesh-Size Control-Concentrated Keypoint**
  - Pick ***a/20*** as the radius of the first circle
  - Pick ***1.5*** for the radius ratio (2nd row/1st row)
  - Use ***5*** or ***6*** for the number of elements around the circumference
  - Use the **Skewed 1/4 pt.** option for the midside node position

- Then use size control along the radial lines emanating from the crack tip (using, for example, 8 elements with a spacing ratio of 2,5) and along the circumference of the semi-circle (say, 12 equal size elements)
- Then mesh the semi-circle with triangular elements (Figure 2)

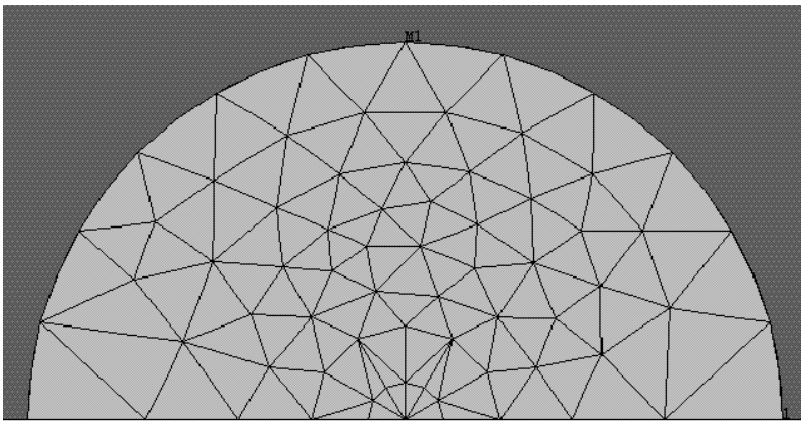


Figure 2. Mesh in crack tip region.

Then create a mesh in the remainder of the domain, using the mesh tool and various levels of refinement. A typical mesh should look like that presented in Figure 3.

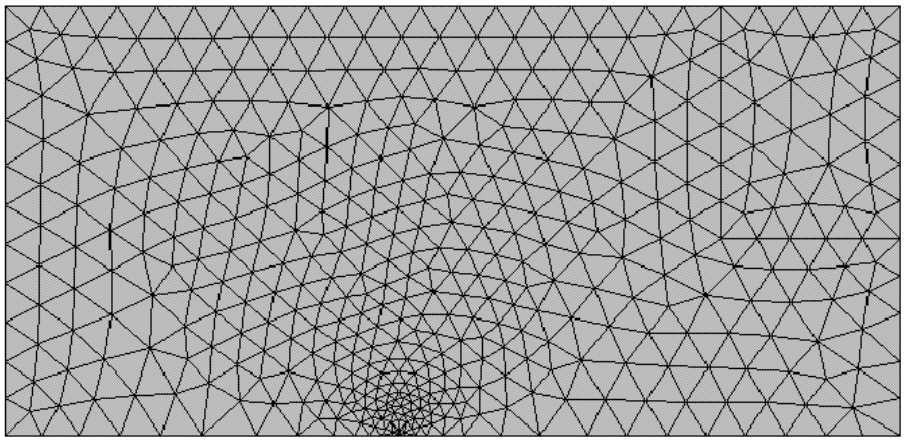


Figure 3. Full mesh.

3) Solution step

Solution >

3.1) Boundary conditions

Apply symmetry bc along the line ahead of the crack tip, zero x-displacement on the point of application of the load, and the vertical load *P* at the keypoint corresponding to the lower left corner of the small rectangle (Figure 4).

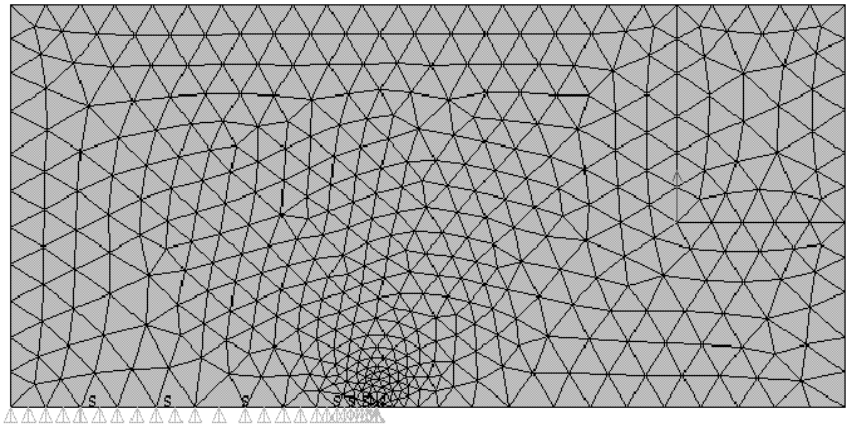


Figure 4. Boundary conditions

3.2) Solution

The solution should take just a few seconds.

4) Post-processing

General Postproc >

4.1) Deformed shape and stress concentration

Contours of the σ_{yy} stress distribution should clearly indicate the presence of a stress concentration at the crack tip (Figures 5 and 6)

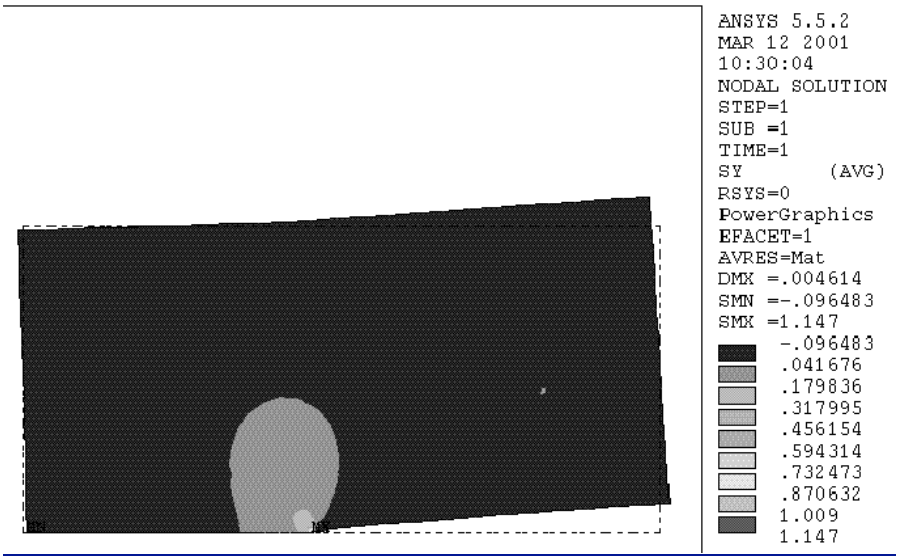


Figure 5. Deformed shape and stress distribution.

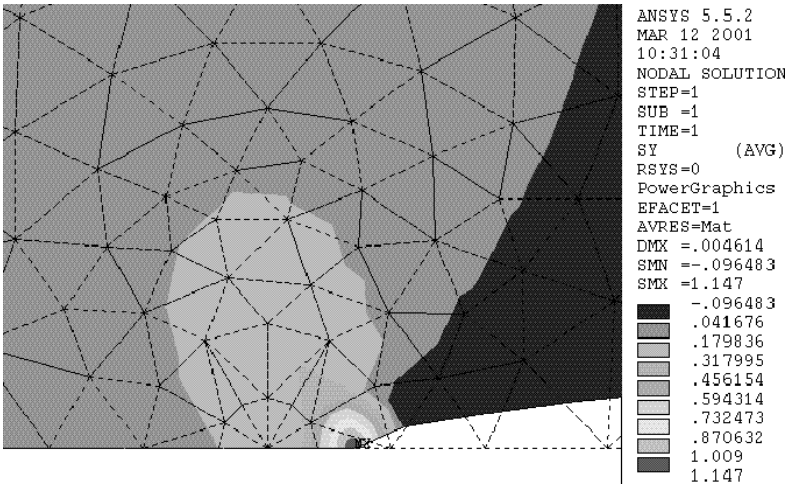


Figure 6. Stress distribution in the crack tip region.

4.2) Extraction of the SIF

As explained in the appendix, we must first define a path in the vicinity of the crack tip. But before doing so, we must define a new coordinate system "pointing ahead of the crack", using the following command:

Work Plane / Local Coordinate System / Create Local CS / By 3 Nodes

- pick the origin first (crack tip node)
- then pick a node along the new x-axis (i.e., a point along the plane of symmetry)
- finally, pick any node in the new x-y plane (i.e., any node off the plane of symmetry)

After defining the new CS, let us define a path with

Path Operators> Define Path> By Nodes+

Then pick successively three nodes behind the crack tip (i.e., along the crack face), with the first one at the crack tip, the second close to the crack tip, and the third one a little further away (you can pick the first three nodes if you want).

Finally, extract the stress intensity factor with **Nodal Calcs> Stress Intensity Factor**, which will create a separate window with the following information:

```
****  CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS  ****

   ASSUME PLANE STRAIN CONDITIONS

   ASSUME A HALF-CRACK MODEL WITH SYMMETRY BOUNDARY CONDITIONS (USE 3 NODES)

   EXTRAPOLATION PATH IS DEFINED BY NODES:       26      36      37
    WITH NODE      26 AS THE CRACK-TIP NODE

   USE MATERIAL PROPERTIES FOR MATERIAL NUMBER     1
    EX =     4000.0       NUXY =    0.35000      AT TEMP = 0.00000E+00

   PRINT THE LOCAL CRACK-TIP DISPLACEMENTS

   CRACK-TIP DISPLACEMENTS:
        UXC = 0.11685E-02  UYC= 0.00000E+00  UZC= 0.78886E-30

      NODE   CRACK FACE    RADIUS       UX-UXC       UY-UYC       UZ-UZC
        26       TIP      0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00
```

```
        36      TOP        0.22500      0.21686E-05  0.13854E-03  0.00000E+00
        37      TOP        0.90000      0.75442E-05  0.27840E-03  0.00000E+00

   LIMITS AS RADIUS (R) APPROACHES 0.0 (TOP FACE) ARE:
        (UX-UXC)/SQRT(R) = 0.34449E-05 (UY-UYC)/SQRT(R) = 0.29160E-03
        (UZ-UZC)/SQRT(R) = 0.00000E+00

 ****  KI =  0.83296    ,   KII =  0.00000E+00,   KIII =  0.00000E+00  ****
```

The value of the SIF is listed at the end of the file. Compare your solution with the table provided in the ASTM standard.

Finally, create a database log file containing the list of the commands you have used. To run a different crack length case, open the log file with your favorite editor, and just change the definition of the parameter *a*. Restart ANSYS and read the input from the log file. The whole problem will be run automatically, including the definition of the three-node path used to compute the SIF. Try it with ***a = 0.55 w*** and compare your solution to the tabulated one.

P_{max} = maximum load that the specimen was able to sustain,
B = thickness of specimen as determined in 8.2.1,
W = width (depth) of specimen, as determined in A3.4.1,
a = crack length as determined in 8.2.2, and
σ_{YS} = yield strength in tension (offset = 0.2 %) (see Test Methods E 8).

A3.5.5 *Calculation of Crack Mouth Opening Compliance Using Crack Length Measurements*—For bend specimens, calculate the crack mouth opening compliance, V_m/P, in units of m/N (in./lb) as follows (see Note A3.2):

$$V_m/P = (S/E'BW)\cdot q(a/W) \quad \text{(A3.4)}$$

where:

$q(a/W)$ = $6(a/W)[0.76 - 2.28(a/W) + (3.87(a/W)^2 - 2.04(a/W)^3 + 0.66/(1 - a/W)^2]$, and

where:

V_m = crack mouth opening displacement, m (in.),
P = applied load, kN (klbf),
E' = Effective Young's Modulus (= E for plane stress, Pa (psi); = $E/(1 - \nu^2)$ for plane strain, Pa (psi)),
ν = Poisson's Ratio, and

S, B, W, and a are as defined in A3.5.3.

NOTE A3.2—This expression is considered to be accurate to within ±1.0 % for any a/W (23). This expression is valid only for crack mouth displacements measured at the location of the integral knife edges shown in Fig. 5. If attachable knife edges are used, they must be reversed or inset to provide the same measurement point location.

A3.5.5.1 To facilitate the calculation of crack mouth opening compliances, values of q (a/W) are given in the following table for specific values of a/W:

a/W	q(a/W)	a/W	q(a/W)
0.450	6.79	0.500	8.92
0.455	6.97	0.505	9.17
0.460	7.16	0.510	9.43
0.465	7.36	0.515	9.70
0.470	7.56	0.520	9.98
0.475	7.77	0.525	10.27
0.480	7.98	0.530	10.57
0.485	8.21	0.535	10.88
0.490	8.44	0.540	11.19
0.495	8.67	0.545	11.53
		0.550	11.87

A3.5.6 *Calculation of Crack Lengths Using Crack Mouth Opening Compliance Measurements*—For bend specimens, calculate the normalized crack length as follows (see Note A3.3):

$$a/W = 0.9997 - 3.95U + 2.982U^2 - 3.214U^3 + 51.52U^4 - 113.0U^5 \quad \text{(A3.5}$$

where:

$U = 1 / \{1 + [(E'BV_m/P)(4W/S)]^{1/2}\}$

NOTE A3.3—This expression fits the equation in A3.5.5 within ±0.01 % of W for $0.3 \le a/W \le 0.9$ (24). This expression is valid only for crack mouth displacements measured at the location of the integral knife edges shown in Fig. 5. If attachable knife edges are used, they must be reversed or inset to provide the same measurement point location.

A4. SPECIAL REQUIREMENTS FOR THE TESTING OF COMPACT SPECIMENS

A4.1 Specimen

A4.1.1 The standard compact specimen is a single edge-notched and fatigue cracked plate loaded in tension. The general proportions of this specimen configuration are shown in Fig. A4.1.

A4.1.2 Alternative specimens may have $2 \le W/B \le 4$ but with no change in other proportions.

A4.2 Specimen Preparation

A4.2.1 For generally applicable specifications concerning specimen size and preparation see Section 7.

A4.3 Apparatus

A4.3.1 *Tension Testing Clevis*—A loading clevis suitable for testing compact specimens is shown in Fig. A4.2. Both ends of the specimen are held in such a clevis and loaded through pins, in order to allow rotation of the specimen during testing. In order to provide rolling contact between the loading pins and the clevis holes, these holes are provided with small flats on the loading surfaces (4). Other clevis designs may be used if it can be demonstrated that they will accomplish the same result as the design shown.

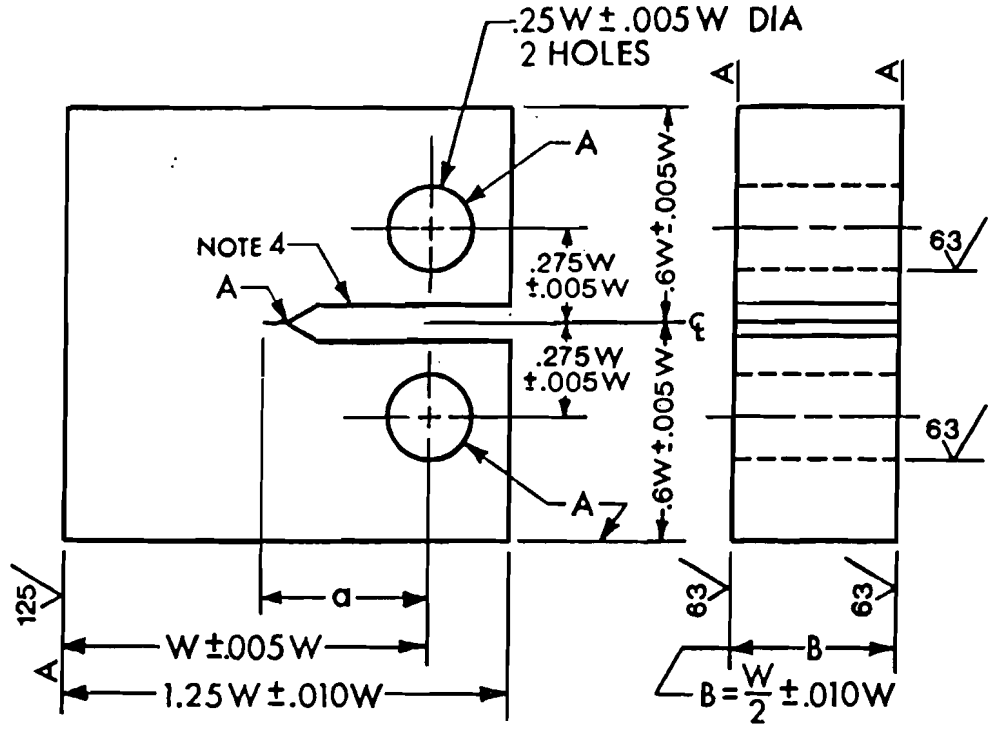
A4.3.1.1 The critical tolerances and suggested proportions of the clevis and pins are given in Fig. A4.2. These proportions are based on specimens having $W/B = 2$ for $B > 0.5$ in. (12.7 mm) and $W/B = 4$ for $B = 0.5$ in. (12.7 mm). If 280 000-psi (1930-MPa) yield strength maraging steel is used for the clevis and pins, adequate strength will be obtained for testing the specimen sizes and σ_{YS}/E ratios given in 7.1.3. If lower-strength grip material is used, or if substantially larger specimens are required at a given σ_{YS}/E ratio than those shown in 7.1.3, then heavier grips will be required. As indicated in Fig. A4.2 the clevis corners may be cut off sufficiently to accommodate seating of the clip gage in specimens less than 0.375 in. (9.5 mm) thick.

A4.3.1.2 Careful attention should be given to achieving as good alignment as possible through careful machining of auxiliary gripping fixtures.

A4.3.2 *Displacement Gage*—For generally applicable details concerning the displacement gage see 6.3. For the compact specimen the displacements will be essentially independent of the gage length up to 1.2 W.

A4.4 Procedure

A4.4.1 *Measurement*—For a compact specimen measure the width, W, and the crack length, a, from the plane of the centerline of the loading holes (the notched edge is a convenient reference line but the distance from the centerline of the holes to the notched edge must be subtracted to determine W and a). Measure the width, W, to the nearest 0.001 in. (0.02



NOTE 1—*A* surfaces shall be perpendicular and parallel as applicable to within 0.002 *W* TIR.

NOTE 2—The intersection of the crack starter notch tips with the two specimen surfaces shall be equally distant from the top and bottom edges of the specimen within 0.005 *W*.

NOTE 3—Integral or attachable knife edges for clip gage attachment to the crack mouth may be used (see Fig. 5 and Fig. 6).

NOTE 4—For starter notch and fatigue crack configuration see Fig. 7.

FIG. A4.1 Compact Specimen C (T) Standard Proportions and Tolerances

mm) or 0.1 %, whichever is larger, at not less than three positions near the notch location, and record the average value.

A4.4.1.1 For general requirements concerning specimen measurement see 8.2.

A4.4.2 *Compact Specimen Testing*—When assembling the loading train (clevises and their attachments to the tensile machine) care should be taken to minimize eccentricity of loading due to misalignments external to the clevises. To obtain satisfactory alignment keep the centerline of the upper and lower loading rods coincident within 0.03 in. (0.76 mm) during the test and center the specimen with respect to the clevis opening within 0.03 in. (0.76 mm).

A4.4.2.1 Load the compact specimen at such a rate that the rate of increase of stress intensity is within the range 30 to 150 ksi·in.$^{1/2}$/min (0.55 to 2.75 MPa·m$^{1/2}$/s) corresponding to a loading rate for a standard (W/B = 2) 1-in. thick specimen between 4500 and 22 500 lbf/min (0.34 to 1.7 kN/s).

A4.4.2.2 For details concerning recording of the test record, see 8.4.

A4.5 Calculations

A4.5.1 For general requirements and procedures in interpretation of the test record see 9.1.

A4.5.2 For a description of the validity requirements in terms of limitations on P_{max}/P_Q and the specimen size requirements see 9.1.2 and 9.1.3.

A4.5.3 *Calculation of* K_Q—For the compact specimen calculate K_Q in units of ksi·in.$^{1/2}$ (MPa·m$^{1/2}$) from the following expression (Note A4.1)

$$K_Q = (P_Q/BW^{1/2}) \cdot f(a/W) \quad \text{(A4.1)}$$

where:

$$f(a/W) = \frac{(2 + a/W)(0.886 + 4.64a/W - 13.32a^2/W^2 + 14.72a^3/W^3 - 5.6a^4/W^4)}{(1 - a/W)^{3/2}} \quad \text{(A4.2)}$$

where:

P_Q = load as determined in 9.1.1, klbf (kN),
B = specimen thickness as determined in 8.2.1, in. (cm),
W = specimen width, as determined in A4.4.1, in. (cm), and
a = crack length as determined in 8.2.2 and A4.4.1, in. (cm).

NOTE A4.1—This expression is considered to be accurate within ±0.5 % over the range of a/W from 0.2 to 1 (12) (13).

A4.5.3.1 To facilitate calculation of K_Q, values of f (a/W) are tabulated below for specific values of a/W.

Compact Specimens

a/W	f(a/W)	a/W	f(a/W)
0.450	8.34	0.500	9.66
0.455	8.46	0.505	9.81
0.460	8.58	0.510	9.96
0.465	8.70	0.515	10.12
0.470	8.83	0.520	10.29
0.475	8.96	0.525	10.45
0.480	9.09	0.530	10.63
0.485	9.23	0.535	10.80
0.490	9.37	0.540	10.98
0.495	9.51	0.545	11.17
		0.550	11.36

3.9 Fracture Mechanics

Cracks and flaws occur in many structures and components, sometimes leading to disastrous results. Several years ago, a commercial airliner flying near the Hawaiian islands suddenly lost the top part of its fuselage. Apparently, microscopic defects introduced in the original fabrication had enlarged by crack propagation over the years until the aluminum skin simply tore apart. This is an example of loss of structural integrity by fracture. The engineering field of fracture mechanics was established to develop a basic understanding of such crack propagation problems.

3.9.3 What is Fracture Mechanics?

Fracture mechanics deals with the study of how a crack or flaw in a structure propagates under applied loads. It involves correlating analytical predictions of crack propagation and failure with experimental results. The analytical predictions are made by calculating fracture parameters such as stress intensity factors in the crack region, which you can use to estimate crack growth rate. Typically, the crack length increases with each application of some cyclic load, such as cabin pressurization–depressurization in an airplane. Further, environmental conditions such as temperature or extensive exposure to irradiation can affect the fracture propensity of a given material.

Some typical fracture parameters of interest are:

- *stress intensity factors* (K_I, K_{II}, K_{III}) associated with the three basic modes of fracture (see Figure 3.9–1)
- *J–integral*, which may be defined as a path–independent line integral that measures the strength of the singular stresses and strains near a crack tip
- *energy release rate* (G), which represents the amount of work associated with a crack opening or closure

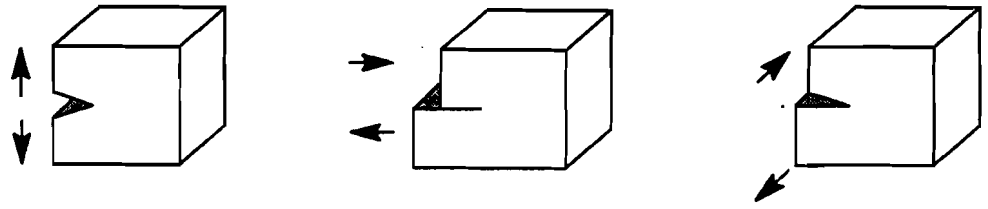


Figure 3.9–11 The three basic modes of fracture

3.9.4 How to Solve Fracture Mechanics Problems

Solving a fracture mechanics problem involves performing a linear elastic or elastic–plastic static analysis and then using specialized postprocessing commands or macros to calculate desired fracture parameters. In this section, we will concentrate on two main aspects of this procedure:

- Modeling the Crack Region
- Calculating Fracture Parameters

See Section 3.2 for details about the general static analysis procedure. See also Section 3.8 for a discussion of structural nonlinearities.

■ Modeling the Crack Region

The most important region in a fracture model is the region around the edge of the crack. We will refer to the edge of the crack as a *crack tip* in a 2–D model and *crack front* in a 3–D model. This is illustrated in Figure 3.9–2.

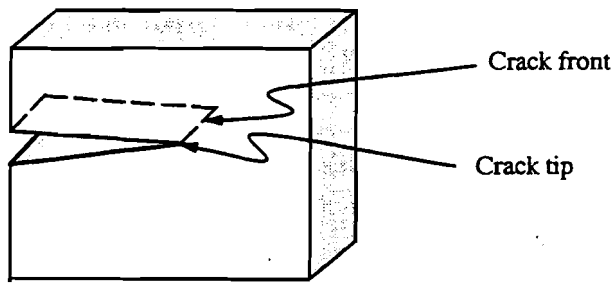


Figure 3.9–12 Crack tip and crack front

In linear elastic problems, it has been shown that the displacements near the crack tip (or crack front) vary as $\sqrt{r}$, where r is the distance from the crack tip. The stresses and strains are singular at the crack tip, varying as $1/\sqrt{r}$. To pick up the singularity in the strain, the elements around the crack tip (or crack front) should be quadratic, with the midside nodes placed at the quarter points. Such elements are called *singular elements*. Figure 3.9–3 shows examples of singular elements for 2–D and 3–D models.

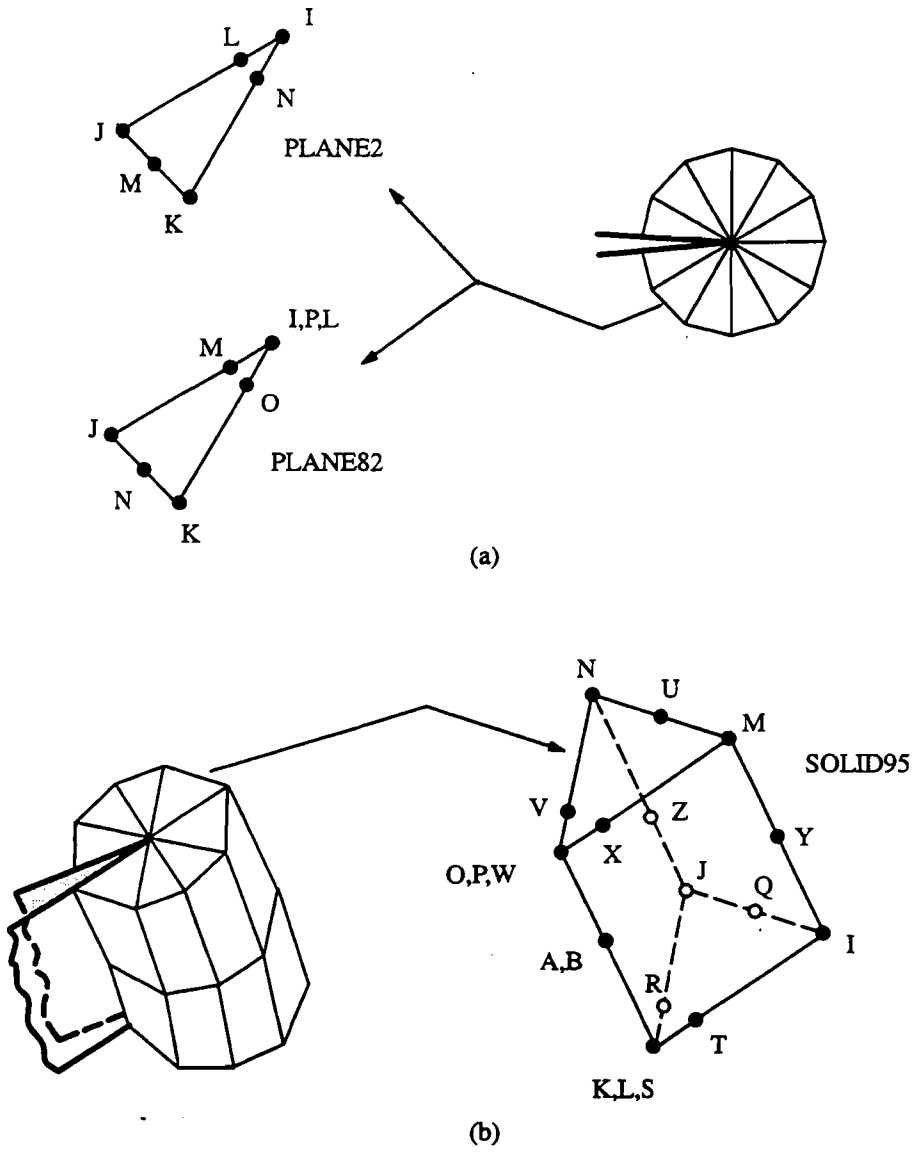


Figure 3.9–13 Examples of singular elements for (a) 2–D models and (b) 3–D models

2–D Fracture Models

KSCON

The recommended element type for a two–dimensional fracture model is PLANE2, the six–node triangular solid. The first row of elements around the crack tip should be singular, as illustrated in Figure 3.9–3(a). The PREP7 **KSCON** command, which assigns element division sizes around a keypoint, is particularly useful in a fracture model. It automatically generates singular elements around the specified keypoint. Other fields on the command allow you to control the radius of the first row of elements, number of elements in the circumferential direction, etc. Figure 3.9–4 shows a fracture model generated with the help of **KSCON**.

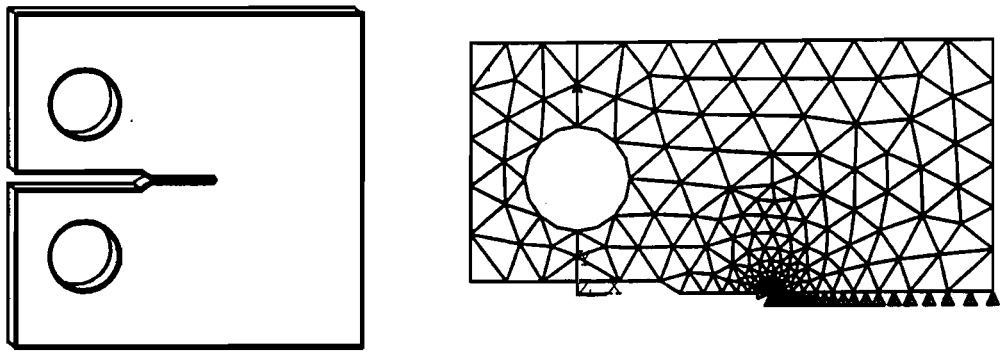


Figure 3.9–14 A fracture specimen and its 2–D F.E. model

Other modeling guidelines for 2–D models are as follows:

- Take advantage of symmetry where possible. In many cases, you need to model only one half of the crack region, with symmetry or anti–symmetry boundary conditions, as shown below.

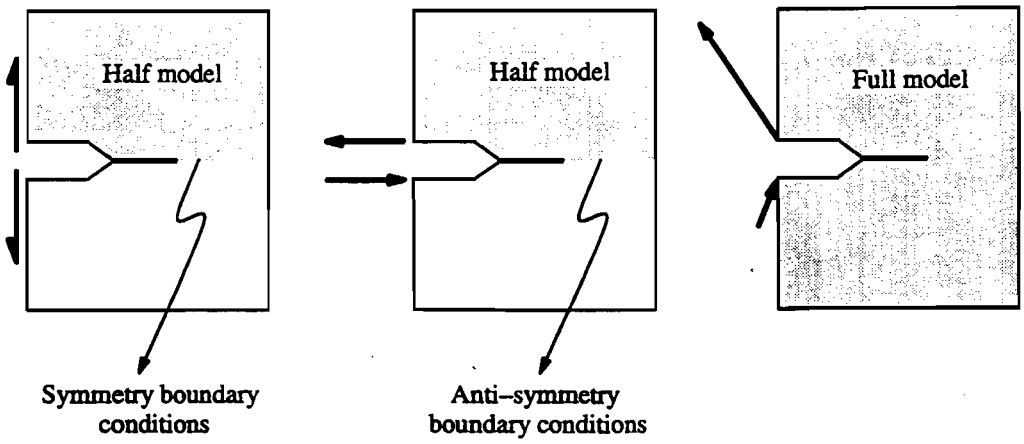


Figure 3.9–15 Taking advantage of symmetry

- For reasonable results, the first row of elements around the crack tip should have a radius of approximately $a/8$ or smaller, where a is the crack length. In the circumferential direction, roughly one element every 30 or 40 degrees is recommended.
- The crack tip elements should not be distorted, and should take the shape of isosceles triangles.

3–D Fracture Models

The recommended element type for three–dimensional models is SOLID95, the 20–node brick element. As shown in Figure 3.9–3(b), the first row of elements around the crack front should be singular elements. Notice that the element is wedge–shaped, with the KLPO face collapsed into the line KO.

Generating a 3–D fracture model is considerably more involved than a 2–D model. The **KSCON** command is *not* available, and you need to make sure that the crack front is along edge IM of the elements.

Other meshing guidelines for 3–D models are as follows:

- Element size recommendations are the same as for 2–D models. In addition, aspect ratios should not exceed approximately 4 to 1 in all directions.
- For curved crack fronts, the element size along the crack front will depend on the amount of local curvature. As a rough guide, you should have at least one element every 15 to 30 degrees along a circular crack front.
- All element edges should be straight, including the edge on the crack front.

■ Calculating Fracture Parameters

Once the static analysis is completed, you can use POST1, the general postprocessor to calculate fracture parameters. As mentioned earlier, typical fracture parameters of interest are stress intensity factors, the J–integral, and the energy release rate.

Stress Intensity Factors

/POST1

The POST1 **KCALC** command calculates the mixed–mode stress intensity factors K_I, K_{II}, and K_{III}. This command is limited to linear elastic problems with a homogeneous, isotropic material near the crack region. To use **KCALC** properly, take the following steps in POST1:

LOCAL
CLOCAL
CS
CSKP
etc.
RSYS

1. Define a local crack–tip or crack–front coordinate system using the **LOCAL** command (or **CLOCAL, CS, CSKP**, etc.), with X parallel to the crack face (perpendicular to the crack front in 3–D models) and Y perpendicular to the crack face, as shown in the following figure. *This coordinate system must be the active model coordinate system [CSYS] and results coordinate system [RSYS] when KCALC is issued.*

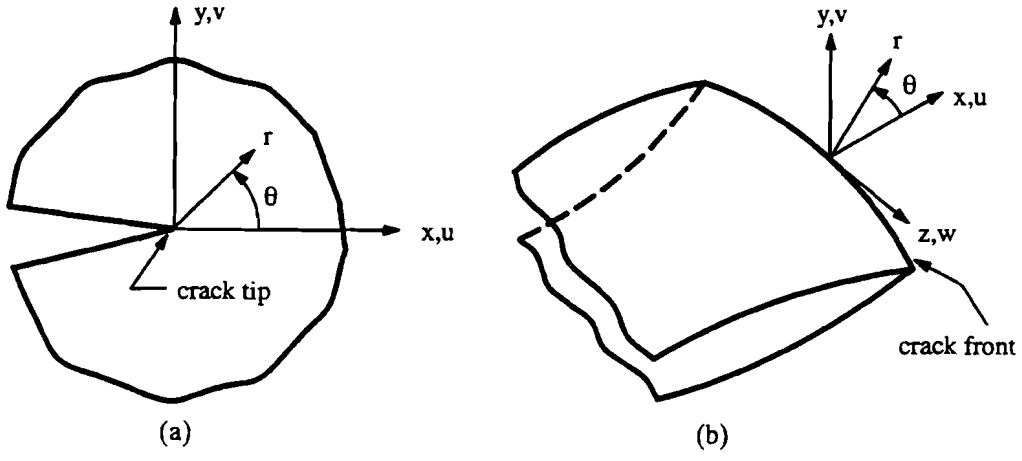


Figure 3.9–16 Crack coordinate systems for (a) 2–D models and (b) 3–D models

LPATH

2. Define a path along the crack face using the **LPATH** command. The first node on the path should be the crack–tip node. For a half–crack model, two additional nodes are required, both along the crack face. For a full–crack model, where both crack faces are included, four additional nodes are required: two along one crack face and two along the other. The following figure illustrates the two cases for a 2–D model.

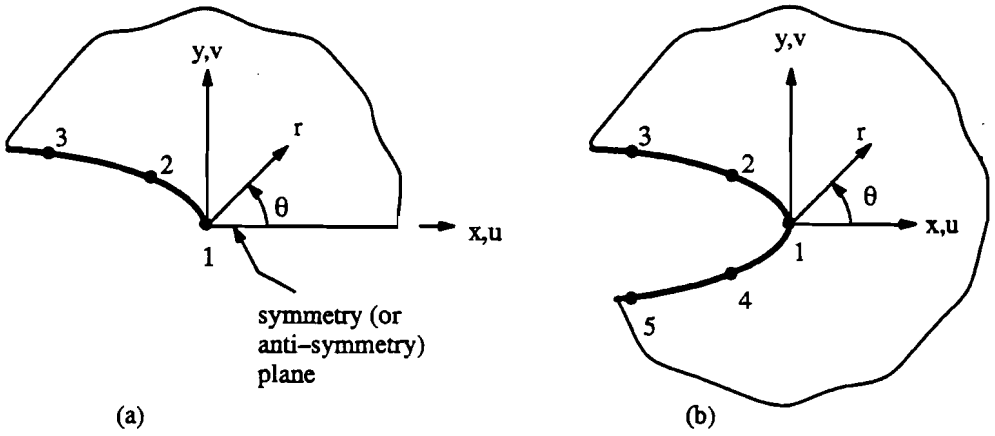


Figure 3.9–17 Typical path definitions for (a) a half–crack model and (b) a full–crack model

KCALC

3. Use the **KCALC** command to calculate K_I, K_{II}, and K_{III}. The *KPLAN* field on the **KCALC** command specifies whether the model is plane–strain or plane stress. Except for the analysis of thin plates, the asymptotic or near–crack–tip behavior of stress is usually thought to be that of plane strain. The *KCSYM* field specifies whether the model is a half–crack model with symmetry boundary conditions, a half–crack model with anti–symmetry boundary conditions, or a full–crack model.

J–Integral

In its simplest form, the J–integral can be defined as a path–independent line integral that measures the strength of the singular stresses and strains near a crack tip. Equation 3.9–1 shows an expression for J in its 2–D form is shown below. It assumes that the crack lies in the global Cartesian X–Y plane, with X parallel to the crack (see Figure 3.9–8).

$$J = \int_\Gamma W dy - \int_\Gamma (t_x \frac{\partial u_x}{\partial x} + t_y \frac{\partial u_y}{\partial y})\, ds \qquad 3.9–2$$

where

Γ = any path surrounding the crack tip
W = strain energy density (i.e., strain energy per unit volume)
t_x = traction vector along x axis = $\sigma_x n_x + \sigma_{xy} n_y$
t_y = traction vector along y axis = $\sigma_y n_y + \sigma_{xy} n_x$
σ = component stress
n = unit outer normal vector to path Γ
u = displacement vector
s = distance along the path Γ

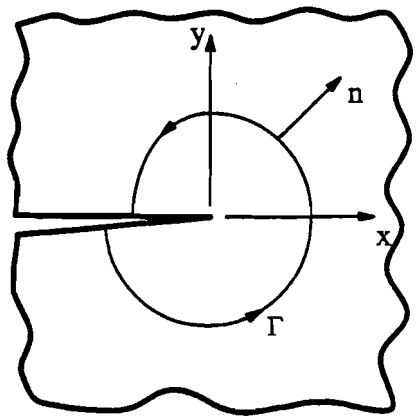


Figure 3.9–18 J–integral contour path surrounding a crack–tip

The steps required to calculate J for a 2–D model are described below:

SET
ETABLE
SEXP
LPATH

1. Read in the desired set of results [**SET**], store the volume and strain energy per element [**ETABLE**], and calculate the strain energy density per element [**SEXP**].

2. Define a path for the line integral [**LPATH**]. Figure 3.9–9 shows examples of such paths.

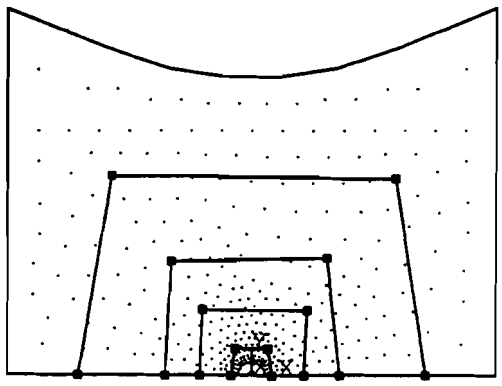


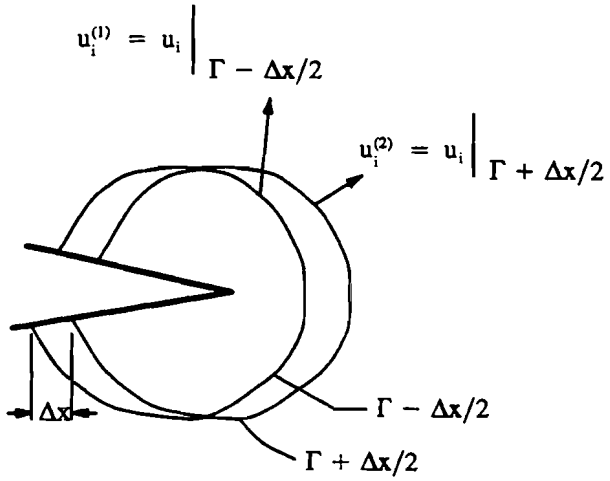
Figure 3.9–19 Examples of paths for J–integral calculation

PDEF
PCALC
PVECT
***GET**

3. Map the strain energy density, which was stored in the element table in step 1, onto the path [**PDEF**], integrate it with respect to global Y [**PCALC**], and assign the final value of the integral to a parameter [***GET**,*Name*,PATH,,LAST]. This gives us the first term of equation 3.9–1.

4. Map the component stresses SX, SY, and SXY onto the path [**PDEF**], define the path unit normal vector [**PVECT**], and calculate TX and TY [**PCALC**] using the expressions shown on with equation 3.9–1.

5. Shift the path a small distance in the positive and negative X directions to calculate the derivatives of the displacement vector ($\partial u_x/\partial x$ and $\partial u_y/\partial y$). The following steps are involved (see Figure 3.9–10):
   - Calculate the distance by which the path is to be shifted, say DX. A rule of thumb is to use one percent of the total length of the path. You can obtain the total path length as a parameter using ***GET**,*Name*,PATH,,LAST,S.
   - Shift the path a distance of DX/2 in the negative X direction [**PCALC**,ADD,XG,XG,,,,–DX/2] and map the displacements UX and UY onto the path [**PDEF**], giving them labels UX1 and UY1, for example.
   - Shift the path a distance of DX in the positive X direction (i.e., +DX/2 from its original position) and map UX and UY onto the path, giving them labels UX2 and UY2, for example.
   - Shift the path back to its original location (a distance of –DX/2) and calculate the quantities (UX2–UX1)/DX and (UY2–UY1)/DX using **PCALC**. These quantities represent $\partial u_x/\partial x$ and $\partial u_y/\partial y$, respectively.



$$\frac{du_i}{dx} \cong \frac{u_i^{(2)} - u_i^{(1)}}{\Delta x}$$

```
*GET,DX,PATH,,LAST,S
DX=DX/100
PCALC,ADD,XG,XG,,,,-DX/2
PDEF,UX1,U,X
PDEF,UY1,U,Y
PCALC,ADD,XG,XG,,,,DX
PDEF,UX2,U,X
PDEF,UY2,U,Y
PCALC,ADD,XG,XG,,,,-DX/2
C=1/DX
PCALC,ADD,C1,UX2,UX1,C,-C
PCALC,ADD,C2,UY2,UY1,C,-C
```

Figure 3.9–20 Calculating derivatives of the displacement vector

6. Using the quantities calculated in steps 4 and 5, calculate the integrand in the second term of J [**PCALC**] and integrate it with respect to the path distance S [**PCALC**]. This gives the second term of 3.9–1.

7. Calculate J according to equation 3.9–1, using the quantities calculated in steps 3 and 6.

You can simplify the J–integral calculations by writing a macro that performs the above operations. (Macros are described in Section 15.3, page 15–7.)

Energy Release Rate

Energy release rate is a concept used to determine the amount of work (change of energy) associated with a crack opening or closure. One method to calculate the energy release rate is the *virtual crack extension method*, outlined below.

In the *virtual crack extension method*, you perform two analyses, one with crack length a and the other with crack length $a+\Delta a$. If the potential energy U (strain energy) for both cases is stored, the energy release rate can be calculated from

$$G = -\frac{U_{a+\Delta a} - U_a}{B\Delta a}$$

where B is the thickness of the fracture model.

Extending the crack length by Δa for the second analysis is quite simple: select all nodes in the vicinity of the crack and scale them in the X direction [**NSCALE**] by the factor Δa. (Note: If you used solid modeling, you will first need to *detach* the solid model from the finite element model [**MODMSH**,DETACH] before scaling the nodes.) The "vicinity of the crack" is usually taken to mean all nodes within a radius of $a/2$ from the crack tip. Also, the factor Δa for node scaling is usually in the range of 1/2 to 2 percent of the crack length.